Quick Incompact3d guide and notes

This is a quick guide on to how to use **Incompact3d** and to know some details of its functioning.

Compiling

It is suggested to create a *build* directory in the same folder of the solver.

```
mkdir build
```

In the *build* directory run

```
cmake ../
```

After that, the *makefile* will be created. It is now possibly to compile the binary file in the build directory with the following command:

```
make -j n
```

where `n` is the number of processors that will be used to compile the program.

Basic functioning

In this section, the main features of the code that can be useful to run simulations are reported.

In Incompact3d:

1. Dimensional Navier-Stokes equations are solved.
2. The Re specified in the input file is in general used to compute the kinematic viscosity as

$$\nu = \frac{1}{Re}$$

   so we are assuming unitary reference length and velocity scales. In case of a channel flow simulation, if constant pressure gradient (CPG) option is enabled, a different way to calculate ν is adopted (see the `parameters.f90` file, Re is based on the centerline velocity of a Poiseuille flow).
3. For a channel flow, the domain dimensions are normalized by the channel half-height h.
4. To evaluate the stretching parameter for the mesh β, a Fortran program is given in the folder *utilities*.
5. Variables are saved on y_p points, that are the faces of the grid elements. On the other hand, y_{pi} points are the centers of the grid elements (i: internal).